

CFD based Dynamic Analysis of Atmospheric Re-Entry Vehicles

M. Korfanty, J. Longo

Institute of Aerodynamics and Flow Technology, Spacecraft Department
Lilienthalplatz 7, 38108 Braunschweig, Germany
e-mail: Marius.Korfanty@dlr.de

A Computational Fluid Dynamics analysis is undertaken to predict static- and dynamic-aerodynamic coefficients for re-entry vehicles. To determine the dynamic behavior of flying vehicles, here is presented a strategy based on the DLR CFD-TAU code with integrated flight mechanics equations in combination with a novel Chimera-Technique. The new Chimera strategy provides also a way to perform grid independent studies when using hybrid meshes. The present results demonstrate the potential of the technique as a major step to remove the classical illness of the force oscillation approach for the prediction of dynamic derivatives on re-entry vehicles.

Introduction

Atmospheric re-entry vehicles like capsules or non-winged lifting-bodies show poor aerodynamic performance. The knowledge of the dynamic response of the vehicle due to perturbations and/or control surface deflections is of major importance, a challenge which is even more severe since these vehicles are exposed to large flight trajectories including strongly varying flight conditions [1]. The concept of a stability derivative is related to the traditional form of equations of motion where the result of a small disturbance from the equilibrium flight condition is described by a linear superposition of contributions caused by the change in various attitude variables and their time rates of change. Even if one has to recognize that a stability derivative is not always a constant but may sometimes be a function of one or more displacement variables, the traditional stability derivatives are constants representing the rate of change of a given aerodynamic coefficient with the variable in question, at a point where the variable itself is zero. The damping derivatives are defined by the rule that each vector component of the force or moment is differentiated with respect to the same corresponding vector component. As example for the longitudinal case, for a free-flying vehicle the variations in the angle of pitch and in the angle of attack can occur independently of each other, and each gives rise to a different distribution of the normal velocity along the vehicle-longitudinal axis. The distribution due to the angle-of-pitch variation varies along the longitudinal axis and intersects zero at the axis of rotation, while the distribution due to angle-of-attack variation is constant along the longitudinal axis. In the case of an oscillation around a fixed axis both variations occurs at the same time and even if the two variables are themselves equal, their effect are different and have to be superimposed. As already indicated, the contribution is equivalent to one due to vertical acceleration.

Wind tunnel experiments, free flight experiments and numerical investigations are the common techniques to examine the dynamic behavior of aerospace vehicles. Because free flight experiments are costly and wind tunnel experiments due to the limited Mach-number and test time not suitable for hypersonic flow, numerical investigations are well adequate for dynamic analysis of hypersonic vehicles. Some existing prediction methods are based on semi-empirical formulas for

the individual vehicle components. This is the DATCOM-type approach. The answers of these methods are rather rough and they often fail to display the critical behavior of a particular configuration. Other methods are based upon linearized potential equation and they use oscillating boundary formulations for the unsteady effects. These methods provide accurate results as long as the aerodynamics is governed by irrotational flow and small perturbation. Many of the existing theoretical methods are restricted to particular geometries such as slender bodies, delta wings, etc. [3]. Most common, however, is the prediction of dynamic behavior by measuring free and forced oscillations in the wind tunnel [2-4]. The extraction of the desired dynamic derivatives is a rather complex process of applying mechanical and aerodynamic sets of equations. These relate the measured unsteady aerodynamic loads to the derivatives by taking into account inertia, elastic deformations and wind tunnel and support interference effects among others. Further, at the end of the nineteen CFD numerical methods for the prediction of dynamic derivatives based on unsteady Navier-Stokes calculations has been successfully applied as demonstrate Fig. 1 from [5]. The numerical approach there employed resembles wind tunnel procedures using forced harmonic motion of the model and transforming the data into the frequency domain via Fourier transformation [6]. By using the time history evolution of the forces and moments along the oscillation periods, a linear approach is defined based on a curve fitting process due to Fourier analysis. The time histories are transformed into sine and cosine-parts, resulting in 2 sets of defining equations where the Fourier coefficients are then evaluated from the traces of the motion parameters by summation formulas instead of the original integrals [7]. For the further data evaluation, only the constant offset and the real and imaginary parts of the first three harmonics of the model oscillation are of interest because with them the original data can be reproduced very accurately, indicating the possibility to assume in most of cases a linear behavior. At least, previous wind tunnel investigations carried out with a delta wing in subsonic flow and a range of angles of attack up to 45 deg. have shown [8] that for pitching and yawing motion the amplitudes of the second harmonic are approximately one fifth of the size of the first harmonic while the amplitudes of the third harmonic are approximately one tenth of the size of the first harmonic.

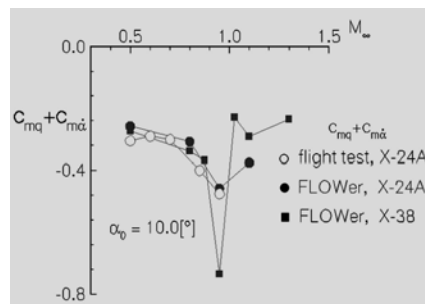


Figure 1: computed dynamic derivatives under force oscillation assumption for a lifting body vehicle.

However, today current designs of re-entry vehicles are calling for flight path with up to 70deg angle of attack. Harmonic force oscillation at that attitude condition may stall the vehicle, presenting areas of massive flow separation and exposing the vehicle to oscillations of large amplitudes which creates complicated flow-field patterns including oscillating separation bubbles or regions alternating between flow-separation and flow-reattachment as is shown in Fig. 2 for the case of the European Intermediate Demonstrator IXV. During the upstroke of the harmonic oscillation, at the nose suction side of the fuselage separating vortices are created growing with

increasing pitch angle. During the downstroke phase of the oscillation, no partial reattachment of the flow occurs. Such a cycle of the harmonic oscillation may also reveal some sort of hysteresis loops. Further, the growing separation bubbles flowing downstream and inducing the effects of rate of change of angle-of-attack are frequency pitch rates dependent. Such flow features put in evidence that for the prediction of dynamic derivatives at huge angle of attack, a methodology like the force oscillation which hardly rely in the assumption of linearization of the problem need to be reviewed since under the here investigated flow conditions this last is a hypothesis difficult to support.

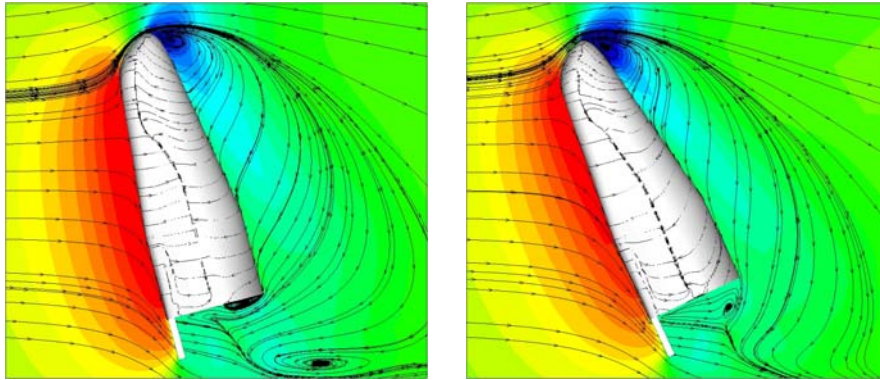


Figure 2: Instantaneous field pressure for $AoA=70$ deg., $Ma=0.8$. Left: upstroke, right: downstroke

To remove the linear restriction on the prediction of dynamic derivatives for re-entry vehicles by means of the force oscillation approach, here is presented a strategy based on the DLR CFD-TAU [6] code with integrated flight mechanics equations in combination with a novel Chimera-Technique. The advantages of this approach are: (i) simulation of arbitrary vehicle motion; (ii) direct determination of dynamic derivatives; (iii) simulation of flight conditions in terms of Mach and Reynolds numbers. Indeed, this new multi-disciplinary approach, involving computational fluid dynamics and flight mechanics allows a direct dynamic analysis of flying vehicles. The current paper demonstrates the applicability of the Chimera-Technique for supersonic and hypersonic flows considering rigid body motion and is a contribution to a wider effort towards the simulation of vehicle fly-control at transonic, supersonic and hypersonic flow conditions.

Fluid / Flight-Mechanic Coupling

Within the DLR Project IMPULSE an independent 6-DOF motion module has been developed and coupled with the CFD-TAU code which enables time accurate coupled simulations of aerodynamic and flight mechanic. It is a stand-alone module with following properties: rigid body motion; resting flat earth (rotation and curvature of Earth is neglected); no restriction on principal axis of inertia. A flow diagram for the CFD/6-DOF simulation process is shown in Fig. 3. The control of the simulation process via a python script enables the possibility of a user intervention in each time step. So it is feasible to realize special boundary conditions, like control surface deflections. Furthermore it integrates the individual disciplines into a unified computational framework.

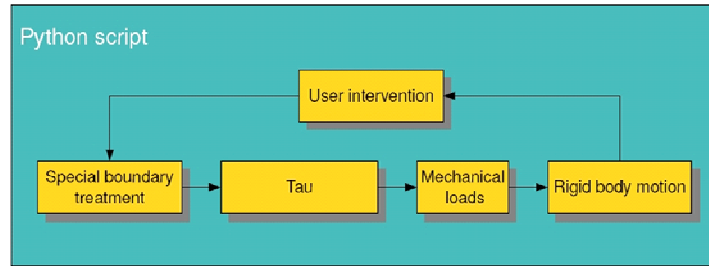


Figure 3: Flow diagram for the CFD / 6-DOF simulation

The 6-DOF module decomposes the rigid body motion into a translation of the center of mass and a rotation about the axis located at the center of mass. The position of the center of mass is updated using Newton's law of motion in the inertial frame. The rotational position of the body is specified using the Euler angles which are updated by integration of the angular velocities. As test example a time accurate coupled simulation is performed for a naca-0012 profile at $M_\infty = 0.5$ which is released from a trimmed position at $\alpha=10^\circ$. The center of pressure is located at the quarter of the body length for a subsonic flow, so a center of rotation position at the tenth part of the body length is taken to assure a statically stable system. The dynamic response of the profile is shown in Fig. 4. As it can be seen, the body performs a dynamic damped motion. Due to rear location of the center of pressure the body gets a backing moment and stabilized itself at $\alpha=0^\circ$. The resulting pressure contours for four different time-steps are shown in Fig. 5.

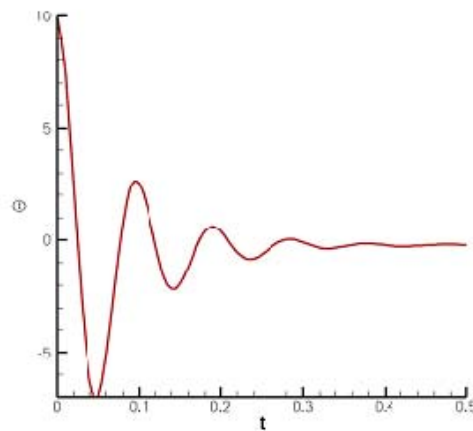


Figure 4: Dynamic response of naca-0012 profile which was released from its trimmed position

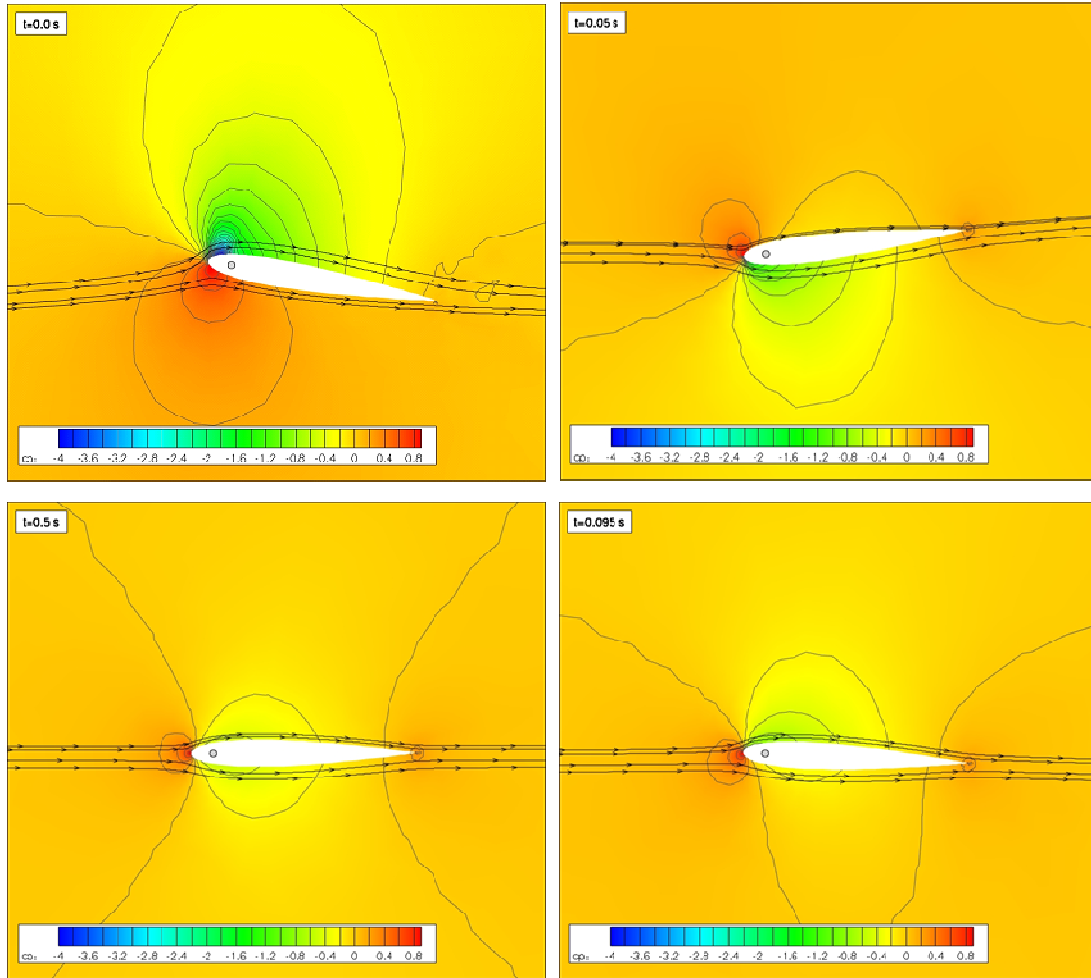


Figure 5: In clockwise direction from top left, pressure contours for different time-steps.

Chimera-Technique

As an essential prerequisite, simulation of maneuvers requires a consideration of vehicles control surfaces. This control surface integration provides a challenge for numerical investigations due to the treatment of the mesh, which must move with the control surface. In order to avoid a new mesh generation for each surface deflection, the technique of overlaid grids, Chimera-Technique, is utilized to integrate the control surfaces. Overset-grid methods are based on subdividing the physical domain into regions that can accommodate easily generated grids. The resulting sets of overlapping grids have to communicate with one another in order to exchange flow information. The original Chimera-Technique developed by Benek [10], accomplishes this exchange using a linear-interpolation of the primitive variables. Unfortunately this method has the disadvantage, that it doesn't maintain conservation across mesh boundaries and needs furthermore an overlapping region of at least two cells. Consequently, degenerations in both accuracy and convergence may result, especially in regions of high gradients. Different investigations using the original Chimera-Technique show a good applicability for subsonic and low supersonic flows [11-12] but for high supersonic and hypersonic cases an artificial shock reflection is generated in the interface of the

grid-boundaries as exemplary demonstrate Fig. 6 for a NACA 0012 airfoil, at Mach number 20, 20° angle of attack.

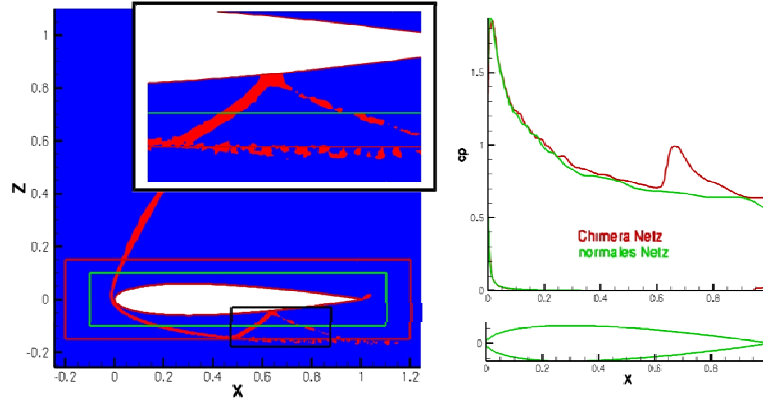


Figure 6: Mach number contour (left) and surface pressure (right) using classical Chimera.

Since the standard Chimera-Technique shows artificial shock reflections in supersonic and hypersonic flows a novel extrapolation boundary condition on the corresponding hole-boundaries is here presented. This new Chimera-Method abstains from an overlapping region and needs only a common surface between the two independent grids. Points on the common surface can possess a displacement. The main idea of the new Chimera-Method is shown in Fig. 7.

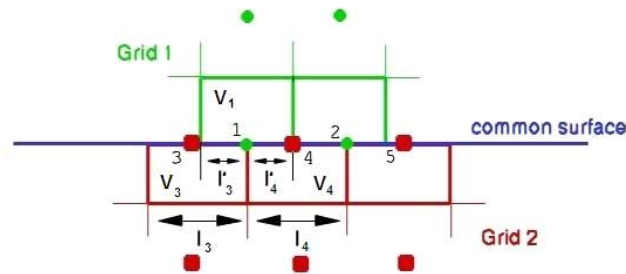


Figure 7: Treatment of boundary points for an extrapolating Chimera-Technique.

In order to get a flow solution at time $t+\Delta t$, following procedure is carried out in each Runge-Kutta step. Both grids are considered separately. The common surface is concerned as an outflow boundary (points on the common surface are extrapolated). To accomplish an exchange of flow information the neighbor points of the different grids have to be coupled. Equation (1) shows the coupling formula of the neighbor points,

$$\mathbf{r}_{W_i} = \frac{\mathbf{r}_{W_i}^{\text{exp}} V_i + \sum_{j=1}^{np} a_j \mathbf{r}_{W_j}^{\text{exp}} V_j}{V_i + \sum_{j=1}^{np} a_j V_j} \quad \text{with } a_j = \frac{l'_j}{l_j} \quad (1)$$

where the subscripts (i,j) represents the cell indices for grid 1 and 2, the superscript *exp* the extrapolated values and *np* the maximum number of neighbor points. $W_{(i,j)}$ represents the Vector of the conservative variables $[\rho, \rho u, \rho v, \rho w, \rho E]$ and $V_{(i,j)}$ the volume of the regarded cells. Due to the fact that the boundary points can posses a displacement, it is possible that the corresponding control volumes are also displaced. This displacement is corrected by the coefficient $a_{(i,j)}$. For W_1 (point 1 in Figure 7) respectively, equation 1 changes to

$$\mathbf{r}_{W_1} = \frac{\mathbf{r}_{W_1}^{\text{exp}} V_1 + \frac{l'_3}{l_3} \mathbf{r}_{W_3}^{\text{exp}} V_3 + \frac{l'_4}{l_4} \mathbf{r}_{W_4}^{\text{exp}} V_4}{V_1 + \frac{l'_3}{l_3} V_3 + \frac{l'_4}{l_4} V_4} .$$

Figure 8 presents once more the NACA 0012 airfoil test case. The surface pressure distribution displayed on the right side of the picture compares results obtained without chimera (reference values) with those obtained using the classical approach (standard) and the new strategy (extrapolation). It turns out the new chimera provides results equivalent to those without chimera.

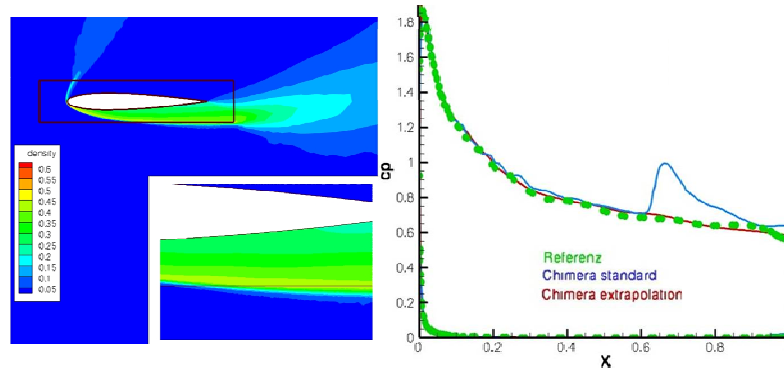


Figure 8: Density contour (left) and surface pressure (right) using the new Chimera technique.

Finally a 2d inviscid $M_\infty = 20$ flow over a 40° ramp is selected to assess the ability of the new scheme to transmit an oblique shock trough a mesh boundary without reflection. Resulting pressure contours are shown in Fig. 9. It can be assessed that the reflection observed for the linearly interpolated solution (left) was eliminated by the new extrapolating method (right). In both cases a similar grid resolution is used.

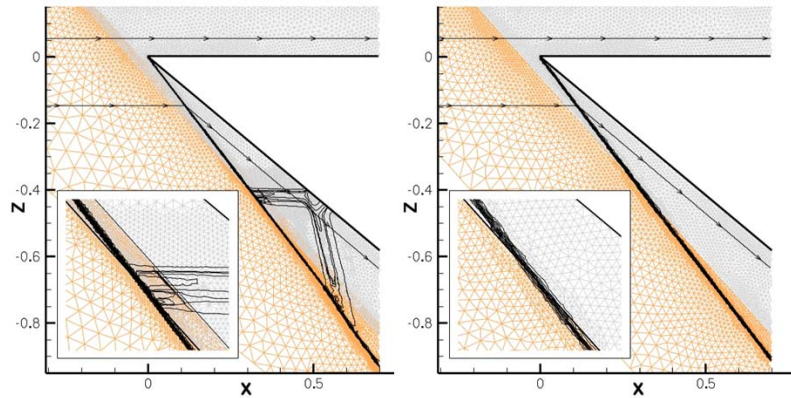


Figure 9: Mach number contour resulting with the old (left) and the new (right) Chimera technique.

Efforts directed to apply the combined Navier-Stokes solver with the integrated flight mechanic equations and the new chimera strategy for the prediction of dynamic derivatives are under way using a Russian Soyuz capsule configuration as example. Here the space vehicle follows an instantaneous pitch-down maneuver embedded in an extreme fine grid which moves together with the vehicle while the applied flow field is resolved in the stationary background grid. Figure 10 present first results in terms of field pressure and streamlines for a transonic motion, Mach 0.8, nominal angle of attack 45° and a 2Hz 5° delta perturbation. The figure shows that the surface pressure resolution is independent of the capsule motion, reflecting in such a way the potential of the new technique for the prediction of dynamic derivative. Current works focuses to obtain similar results under supersonic and hypersonic flow conditions.

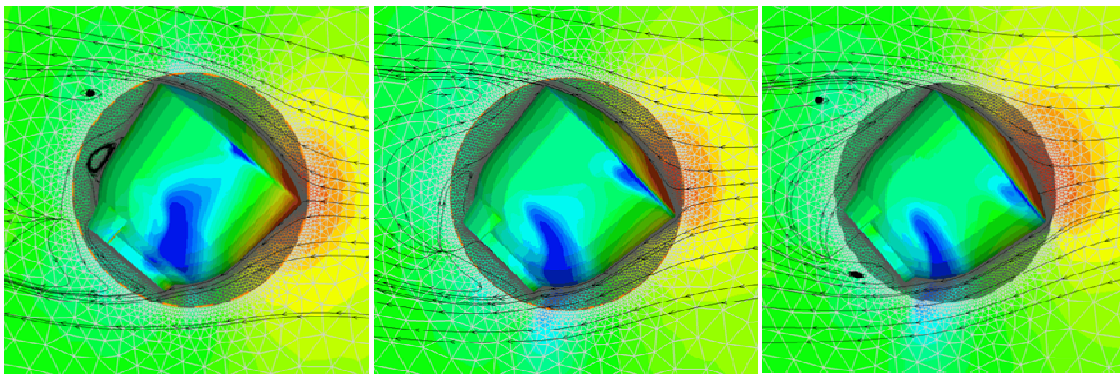


Figure 10: From left to right instantaneous snapshots of a pitch-down maneuver with the Soyuz capsule at $M=0.8$, $AoA=45^\circ$, 2Hz. Pressure contours and streamlines.

Conclusions

The present work is a contribution to a wider effort towards the simulation of vehicle fly-control at transonic, supersonic and hypersonic flow conditions. A Computational Fluid Dynamics analysis is undertaken to predict static- and dynamic-aerodynamic coefficients for re-entry vehicles. To determine the dynamic behavior of flying vehicles, here has been presented a strategy based on

the DLR CFD-TAU code with integrated flight mechanics equations in combination with a novel Chimera-Technique. The new Chimera strategy provides also a way to perform grid independent studies when using hybrid meshes. While the present results still can be considered as preliminary, transonic up to supersonic dynamics motions have been successfully verified, demonstrating the potential of the technique as a major step to remove the classical illness of the force oscillation approach for the prediction of dynamic derivatives on re-entry vehicles.

References

- [1] Kirsten, P. W. A Comparison and Evaluation of Two Methods of Extracting Stability Derivatives from Flight Test Data. AGARD CP 172, 1992, pp.1-18 to 18-2.
- [2] Orlik-Rückemann, K.-J. Techniques for Dynamic Stability Testing in Wind Tunnels. ICAS Paper ICAS-80-7.1, 1980.
- [3] Fuchs, H. Prediction of Dynamic Derivatives. AGARD CP 451, 1979, pp. 6-1 to 6-15.
- [4] Determann, O.; Heege, K. Erweiterung der Ausrüstung und Windkanalerprobung der Mobilen Oszillierenden Derivatwaage MOD. BMFT LFW 7701 I/3 Teil I und II 1079, Institut für Flugmechanik, TH- Darmstadt.
- [5] Giese, P.; Heinrich, R.; Radespiel, R. Numerical Predictions of Dynamic Derivatives for Lifting Bodies with the Navier-Stokes Solver FLOWer. Notes on Numerical Fluid Mechanics, Vol. 70, Vieweg Verlag Braunschweig.
- [6] Wünnenberg, H.; Friedrich, H.; Von Meier, U.; Munser, H.-J. Determination of Stability Derivatives from Flight Test Results Comparison of Five Analytical Techniques. AGARD CP-172, 1992, pp. 10-1 to 10-12.
- [7] Löser, Th.; Bergmann, A. Development of the Dynamic Wind Tunnel Testing Capabilities at DNW-NWB. AIAA Paper AIAA-2003-453, 41st Aerospace Sciences Meeting and Exhibit, Reno, Nevada, Jan. 6-9, 2003.
- [8] Löser, Th. Dynamic Force and Pressure Measurements on an Oscillating Delta Wing at Low Speeds. DLR IB 129-96/9, March 1996.
- [9] Mack, A.; Hannemann, V. Validation of the Unstructured DLR-TAU-Code for Hypersonic Flows. 32nd AIAA Fluid Dynamics Conference, St. Louis (USA), 2002.
- [10] Benek J. A.; Steger J. L.; Dougherty F.C.. A flexible grid embedding technique with application to the Euler equation. AIAA Paper 1983-1944, 1983.
- [11] Schwarz, Th.; Khier, W.; Raddatz, J. RANS Simulation of the Unsteady Flow around the Complete BO-105 Wind Tunnel Model. 7th ONERA-DLR Aerospace Symposium, Toulouse, France, 4.- 6. October 2006.
- [12] Gomez, R. J.; Ma, E. C. Validation of a Large Scale Grid System for the Space Shuttle Launch Vehicle. AIAA Paper 1994-1859, 1994.